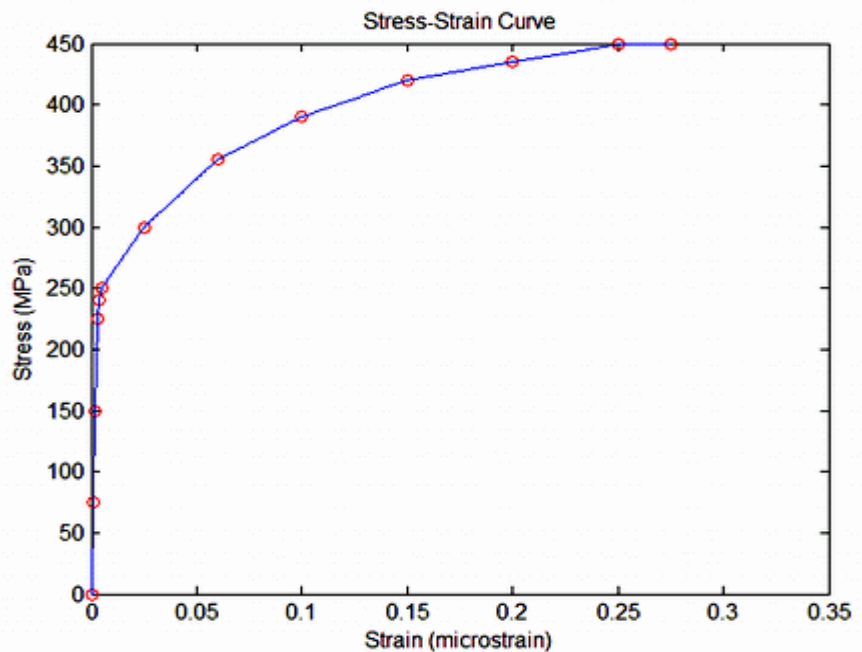
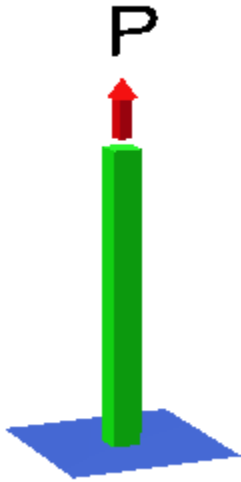


NonLinear Materials

Introduction

This tutorial was completed using ANSYS 7.0. The purpose of the tutorial is to describe how to include material nonlinearities in an ANSYS model. For instance, the case when a large force is applied resulting in stresses greater than yield strength. In such a case, a multilinear stress-strain relationship can be included which follows the stress-strain curve of the material being used. This will allow ANSYS to more accurately model the plastic deformation of the material.



For this analysis, a simple tension specimen 100 mm X 5 mm X 5 mm is constrained at the bottom and has a load pulling on the top. This specimen is made out of an experimental substance called "WhoKilledKenium". The stress-strain curve for the substance is shown above. Note the linear section up to approximately 225 MPa where the Young's Modulus is constant (75 GPa). The material then begins to yield and the relationship becomes plastic and nonlinear.

ANSYS Command Listing

```
finish
/clear

/prep7                ! Enter Preprocessor
k,1,0,0              ! Keypoints
k,2,0,100
l,1,2                ! Line connecting keypoints

ET,1,LINK1          ! Element type
R,1,25              ! Area of 25
```

```

MP,EX,1,75000           ! Young's modulus
MP,PRXY,1,0.3          ! Poisson's ratio

TB,MELA,1,1,12,        ! Create a table of 12 data points
                       ! to map the stress-strain curve

TBPT,,.001,75          ! Data points
TBPT,,.002,150
TBPT,,.003,225
TBPT,,.004,240
TBPT,,.005,250
TBPT,,.025,300
TBPT,,.06,355
TBPT,,.1,390
TBPT,,.15,420
TBPT,,.2,435
TBPT,,.25,449
TBPT,,.275,450

ESIZE,5                ! Element size 5
LMESH,all              ! Line mesh all lines

FINISH
/SOLU                  ! Enter solution phase

NLGEOM,ON              ! Nonlinear geometry on
NSUBST,20,1000,1      ! 20 load steps
OUTRES,ALL,ALL         ! Output data for all load steps
AUTOTS,ON              ! Auto time-search on
LNSRCH,ON              ! Line search on
NEQIT,1000            ! 1000 iteration maximum

ANTYPE,0               ! Static analysis
DK,1,all               ! Constrain keypoint 1
FK,2,FY,10000         ! Load on keypoint 2

SOLVE
FINISH

/POST1                 ! Enter post processor
/ESHAPE,1              ! Show element shape
PLNSOL,U,Y,0,1        ! Plot deflection contour

FINISH

/POST26                ! Enter time history
RFORCE,2,1,F,Y         ! Reads force data in variable 2
NSOL,3,2,U,Y           ! Reads y-deflection data into var 3
XVAR,2                 ! Make variable 2 the x-axis
PLVAR,3
/AXLAB,Y,DEFLECTION    ! Changes y label
/AXLAB,X,LOAD          ! Changes X label
/REPLOT

```